Conference on Modelling Fluid Flow (CMFF'25)
The 19th International Conference on Fluid Flow Technologies
Budapest, Hungary, August 26 - August 29, 2025

A LOOK BACK ON 30 YEARS OF TURBOMACHINERY RESEARCH IN EUROPE.

Gérard BOIS[1]

[1] Corresponding Author. Emeritus Professor ENSAM. Fluid Mechanics Lab. Lille (LMFL).
University of Lille, CNRS, ONERA, Arts et Métiers Institute of Technology, Centrale Lille Institut, UMR 9014-LMFL - Kampé de Fériet, F-59000, Lille, France. E-mail: Gerard.bois@ensam.eu

ABSTRACT

This document aims to give a personal and therefore partial view of what has been learned, performed and achieved during the past 30 years in rotating machinery research fields and what are the possible new research targets in this specific domain. Numerical techniques will not be addressed here; the present paper comes from a CFD end-user and not from a CFD expert. This is the reason why physical aspects are rather highlighted here.

It does not claim to cover all aspects of the work carried out in all types of rotating machinery, but to look at specific research aspects that have contributed to a better understanding of turbomachinery aerodynamic features thanks to basic configuration studies, advanced experimental techniques and powerful data acquisition treatments.

At the end of the present document, the use of artificial intelligence to improve decision-making is put into perspective.

Keywords: experimental, numerical, research, turbomachinery.

PREFACE.

I would like to express my gratitude by dedicating some words to the memory of my former teachers and mentors from the von Karman Institute (VKI) and more particularly that of Claus Sieverding (passed on January 2024) that should have prepared this document with me on the kind invitation from Prof Janos Vad.

My respectful thoughts also go both to Prof. Franz Breugelmanns and Prof. Jacques Chauvin who founds the Turbomachinery Department of VKI.

Left: Claus SIEVERDING
Middle: Franz.A.E.BREUGELMANNS
Right: Jacques CHAUVIN

1. INTRODUCTION

It is obvious to say that the way turbomachinery's aerodynamic research has evolved during the past 30 years is strongly related to the emergence of improved numerical simulations starting from the 1980's and the development of advanced experimental techniques, data acquisition and reduction procedures.

The analysis procedure deduced from CFD flow capturing in single and multistage row configurations including unsteady behavior has been strongly transformed compared with steady 2D thinking that generally took place before the 90's.

With the development of high-performance computing resource and commercial or in house software, full 3D calculations in 2D straight and annular cascades, single stage and multistage configurations of increasingly complex architectures have been made possible both for ground and aerospace applications.

However, the use of numerical and experimental means has been carried out differently depending on the application and market needs. Single stage configurations like low-speed fans, hydraulic machines, wind and marine turbines, propellers have different design and mechanical requirements than high speeds multistage compressors and turbines, turbochargers and space propulsion systems. Thus, different priority strategies and research targets have been set up for high-speed multistage machines (with high temperature and pressure levels with strong gradients, cooling systems and controls) and for hydraulic machineries subject to phase modification in stage components.

Nevertheless, similar attempts and achievements aiming at robust design concepts have been forced because of increasing end-users demands and expectations, manufacturing evolutions and political and economic constraints.

2. CONTEXT

Tasks chairing between those involved in fundamental research and those in applied research was significantly modified starting around the 1980's, a period for which people have essentially sought to obtain a good restitution of global or local performances from CFD without correctly check the validity and the limitations of their approaches or naturally preferred to only show their best results. Later, comparisons with experiments have made it possible to detect the importance of certain phenomena hitherto considered negligible in terms of their consequences. This was performed in association with extended close collaborations with research institutions, manufacturers and end-users, better CFD's accuracy and modelling re-enforced by new machine architectures and manufacturing techniques evolutions.

This was greatly facilitated by proactive actions of partnerships and exchanges promoted by the European Commission Research Programs including sustainable development policies. Competition between machine manufacturers was evolving towards closer collaboration in view of the increase in costs and the duration of research and development activities.

As emphasized by Cumpsty, [1]: "It *was an exceptional period of intellectual competition carried out with great openness and courtesy. Those involved recall it as a stimulating and enjoyable episode a quarter century later".*

3. TECHNICAL AND SCIENTIFIC PUBLICATIONS IN EUROPE

A rapid statistical overview on turbomachinery field publications per country, given on figure 1, can be seen as reflecting most of manufacturers' implantations in each European country. Significant differences can be noticed if scientific publications are classified into categories (figures 2a to 2f). These differences should be related to the energy policies and resources of each country as shown in figure 3.

Note. Publications for wind turbine are not involved in the present document (Main wind turbine manufacturers are in Denmark-Nordex SE and Vestas Wind Systems S/A and in Spain Siemens Gamesa Renewable Energy, S.A).

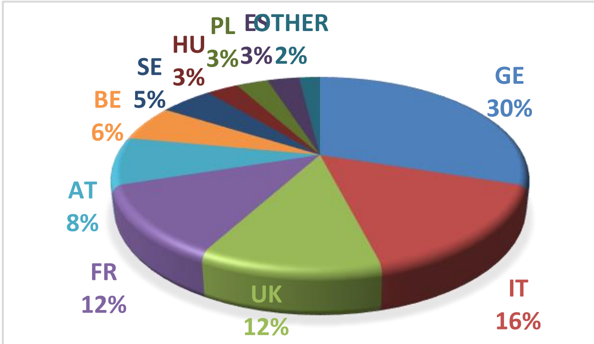


Figure 1. Turbomachinery aerodynamic publications' distribution per countries (1995-2025).

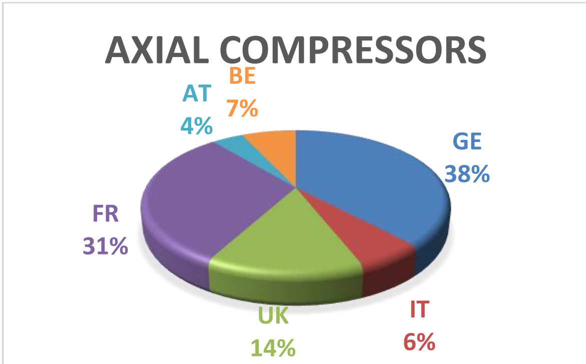


Figure 2a. Distribution for axial compressors

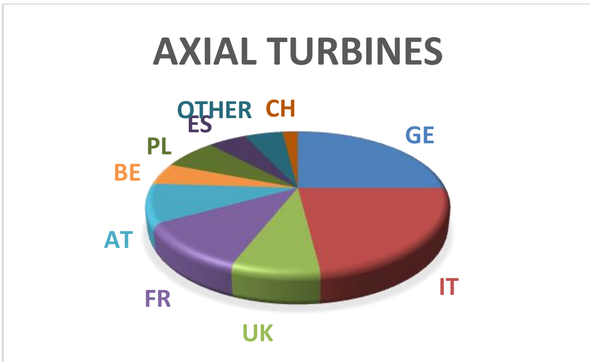


Figure 2b.Distribution for axial turbines.

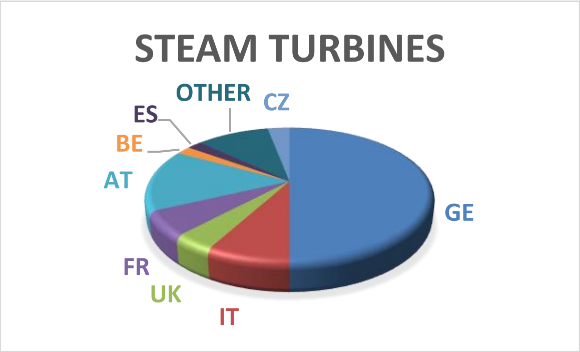


Figure 2c. Distribution for steam turbines

Figure 2d. Distribution for fans.

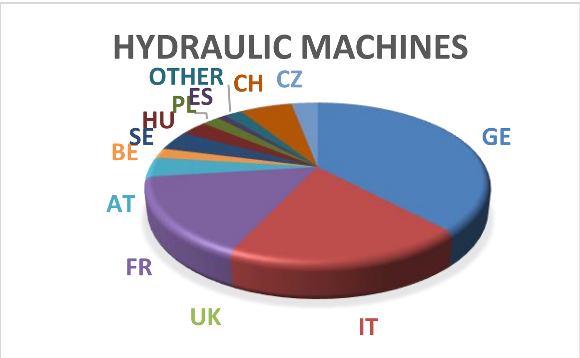


Figure 2e. Distribution for hydraulic machines

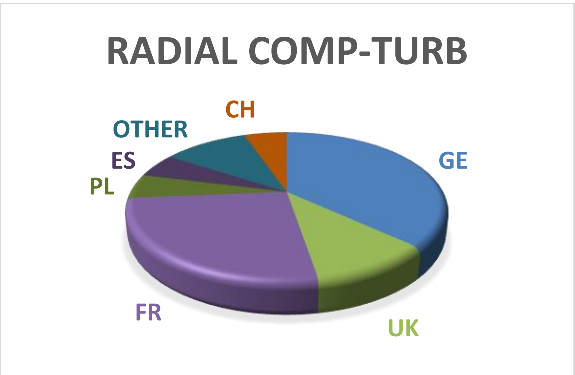


Figure 2f. Distribution for radial machines

Figures 2a to 2f. Research publications per topics.

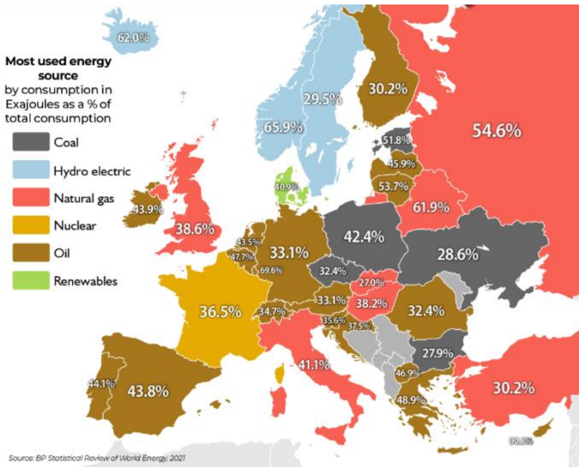


Figure 3. Main energy sources by country.

4. TURBOMACHINERY DESIGN PROCEDURE.

From the beginning, rotating machinery development is strongly based on a well-organized relationship between geometry definition, internal flow modelling, flow behavior understanding and performance analysis**.** Each step used models that can be classified according to the degree of complexity involved in the flow description. They can be divided into four distinct categories: statistical, one-dimensional (1D), two-dimensional (2D), and three-dimensional (3D) models.

During the last 25 years, thanks to larger computer resources and CFD improvements, inviscid and viscous approaches have been successfully used to analyse specific geometry components along with experimental validation both in 2D and 3D cases. One of the best way to analyse existing geometry concerns the pressure distribution in the blade to blade (tangential) and meridional directions either for single component designs and for the complete machine. Figure 4 shows the classical design chart for such purpose. The dates appearing on figures 4 and 5 indicate the most relevant CFD design steps associated to the accurate disposal of physical models related to turbomachinery aerodynamics research that are also recalled below:

a) Integrations of experimental results: up to the 70's

b) Model derivations: 1970-1990

c) Simulations

Up to 80's: 2D inviscid steady and Q3D
1990: 3D NS Steady multistage. Mixing plane assumption.
1995-2000: 3D Unsteady multistage.

d) Data science

2010: Optimization- Robust designs
2020: Artificial Intelligence (AI).

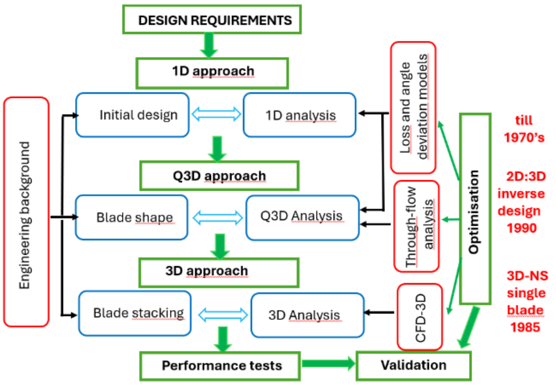


Figure 4. Turbomachinery design/analysis procedure

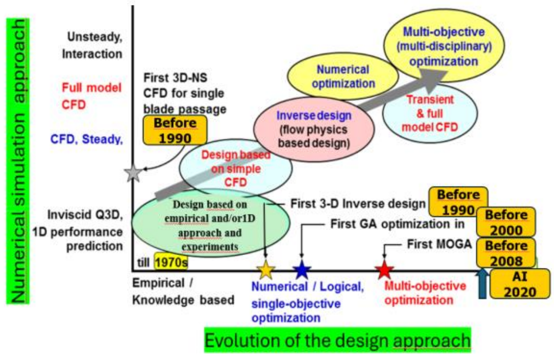
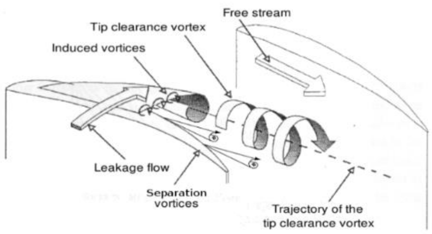


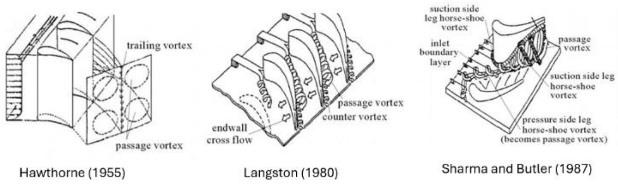
Figure 5. Design approach evolution along with increasing CFD modelling.

4.1. 3D flows in 2D cascades.

One of the most well-known phenomena that encountered numerous experimental and numerical studies, is related to what is commonly called "passage secondary flows" and "tip leakage flows" both for low-speed machines and transonic or supersonic ones. Basic research aiming at deep understanding on physical phenomena were initially performed in 2D cascades, then in annular cascades (see reference list [2] to [15)] for some contributions on this topic both for turbines and compressors.



Tip leakage vortex



Reference [2]. **Reference [3].** **Reference [4].**

Figure 6. 3D flow description in 2D cascades from different authors.

Starting with such an old story is directly linked to the fact that 2D blade-to blade surface remains relevant for the analysis for the turbomachinery community both in direct and inverse modes.

3D effects that were initially called "secondary flows" making some confusions about their real relative impact on what was supposed to be the main core flow in terms of pressure change and turning

- Secondary flows effects cannot be considered a kind of small perturbation for low aspect ratio for example.
- High loadings are responsible for greater interactions between main core flow and viscous flow developments.
- Vortex generation and development creates unsteady phenomena which strength and spreading are associated with specific frequencies (Strouhal number) and time-frequency patterns.
- Specific local flow features like, corner separation, corner stall and resulting mixing must be involved in design criteria and loading limitations.
- They can be affected by additional forced phenomena by adjacent steady or rotating components.

Several experimental and 3D numerical investigations continue to be performed in such basic 2D linear or annular cascades including passive and active flow control, vortex generators, tandem configuration, wall flow injection, film cooling, heat transfer measurements and so on….

4.2. Throughflow approach.

One must remember that, between 1970 and 1990, throughflow methods were currently used because they provide a reasonable trade-off between speed and accuracy. They remain still useful for performing parametric studies early in the conceptual and preliminary design phases of multistage compressor and turbines.

The throughflow methods initially rely on the "quasi-three dimensional (Q3D) "simplification that solves azimuth-averaged flow in the meridional plane as first proposed by Wu in 1952 [16].

When steady state assumption is chosen, important flow aspects like the span-wise transport within the flow path (also called "radial mixing") is strongly under-predicted because of the circumferential averaging operation of flow properties at row interfaces (some aspects related to these effects are presented in section 7.2). The only way to correctly evaluate this key point is to perform an unsteady calculation. However, due to the high blade number and stages, such an approach has a computational cost not always suitable for industrial purposes. Currently, only the steady-state simulation can fit in a frequently used design chain. A "radial mixing" prediction, enhancing turbulent viscosity can promote span-wise diffusion and improve the radial mixing prediction of the steady approach. In this respect, large amount of modelling set-ups and improvements has been proposed by numerous researchers to recover part of the information that had been neglected and/or lost using the two-surfaces (meridional and blade-to-blade) coupling procedure. Some of the pioneers' work achievements are listed here after, mainly because their results can be still used to enrich data-based links for future analysis: Smith 1969 [17], Kerrebrock and Mikolajczak 1970 [18], Bosman and Marsh 1974 [19], Hirsch and Warzee 1976 [20], Denton 1978 [21], Denton and Singh 1979 [22], Adamczyk [23] 1985 and Jennions and Stow 1986 [24]

Extension to 3D simulations in rotor-stator and sub-systems components took place progressively, as shown on figure 7, when boundary conditions treatments and flow property transfers between single element, stages or sub-systems became more rigorous including unsteady effects that cannot be separated from turbomachinery applications (see section 7).

In case of multistage configurations, one the most challenging task for designers is to stick with stage matching or even between blade rows. This matching is based on corrected mass flow evaluation which depends on pressure rise and on blockage that are themselves depending on the 3D flow characteristics. This must be evaluated and modelled not only for design point, but for the whole flow range and rotational speeds when compressibility effects are important. This was clearly addressed by Dawes [25] and Cumpsty [1].

.

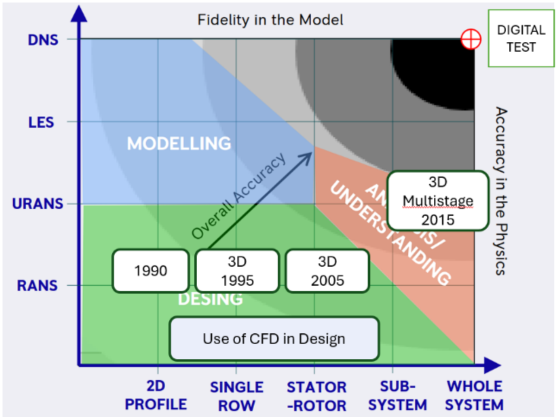


Figure 7. Numerical design and analysis evolutions.

Annulus gaps and corresponding leakage effects may strongly modify redial equilibrium, not only locally but in the entire radial flow path as demonstrated by Shabbir et al [26], in a high-speed rotor and Welborn et al. [27] for a shroud stator cavity. Their effects cannot be included just through a local mean additional flow rate or blockage coefficient but need to be studied by fully 3D calculation. Local geometrical mismatch can also make wrong experimental interpretation (see Escuret et al [28]). Because of increasing blade loading, strong radial gradients, increasing inlet temperatures and cooling effects, new engine architectures and so on, key points have been addressed by several authors like radial mixing evaluation (see [29] for a low aspect ratio rotor), tip clearances, leakages, wall boundary layer and interstage interactions (Adamczyk [30]). They have contributed to include more efficient methods by setting up tuning models, with the help of steady and unsteady simulation results starting from the early 80's. However, one must always be aware of errors that still may arise from different approximations as explained by Denton [31] because full 3D URANS calculations were too time consuming during this period.

Blade loading is a consequence of wall pressure distribution that depends on 3D effects which may be approached either by inviscid simplified hypothesis or viscous modelling. The control of the defined aerodynamic limitations was possible through the development of optimisation techniques that offers better capability to designers to:

- explore and determine the relative influence between several geometrical and flow parameters in single and multi-stage configurations.
- explore the effects and consequences due to leaning, bending and sweeping for the whole range of operation for single components

Some examples are given in the next section to illustrate how design procedures have evolved during this period.

5. EXAMPLES OF 2D/3D BASED APPROACH AND THEIR LIMITS.

These cases have been initially studied mainly based on inviscid considerations even before the 80's. The designs were then improved thanks to viscous steady analysis that took place in the 90's.

5.1. Example 1. 3D shaped of intake elbow and volute designs for pumps and compressors.

Early in1965, Pinckney, S. [32] tried to redesign both non-rotating intake volute to improve overall performances through a non symertic inlet fixed nose to overcome the swirling flow created by the elbow shape (see Figure 8a). Other ideas on the same topic were proposed by Neumann [33] for a tangential volute and by Flathers et al. [34] by introducing fixed non axisymmetric blades (see respectively Figures 8b and 8c).

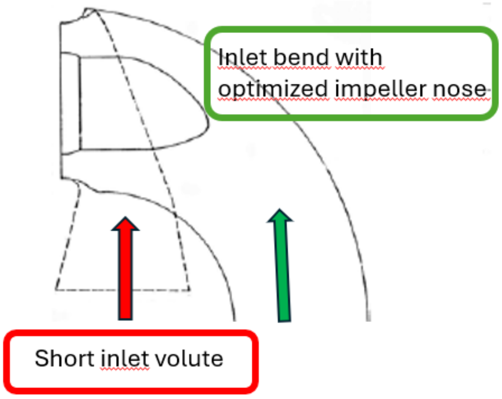


Figure 8a. Non axisymetric inlet with optimized impeller nose.

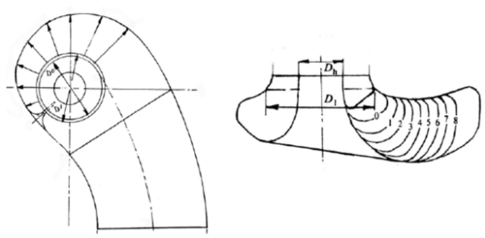


Figure 8b Bladeless tangential inler volute. Extracted from [33],

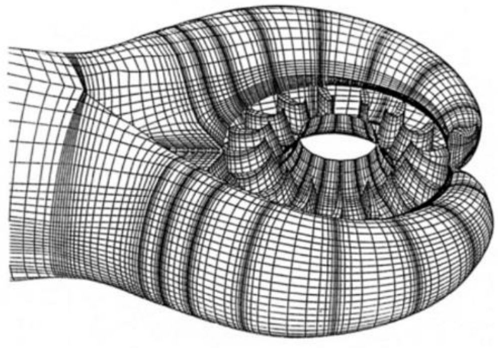


Figure 8c. Inlet volute with inlet guide vanes. Extracted from [34],

During the 90's and in early 2000's, specific analysis on flow in volutes were performed and published by Ayder and Van den Braembussche [35] to [37] and Van den braembussche et al. [38]. A complete revue on volute flow analysis can be also found from the same author in [39].

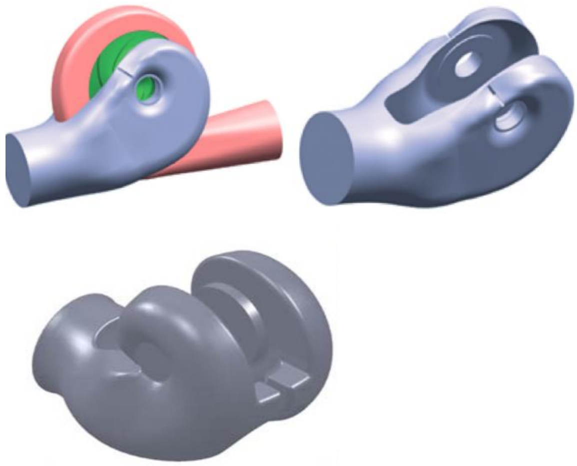


Figure 9. Examples of optimized geometries of inlet and outlet volutes and inlet chamber of a double suction pump..

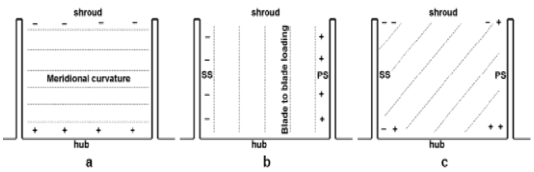
Because inlet and outlet volute analysis cannot be dissociated from the rotating parts, inverse approach techniques were implemented both for fixed and rotating parts designs in mixed and centrifugal pumps and compressors by taking, for example, non uniform inlet conditions (Zangeneh [40], [41], [42] and Demeulenaere and Van den Braembussche [43]).

5.2. Example 2. Centrifugal Impeller blade lean effects.

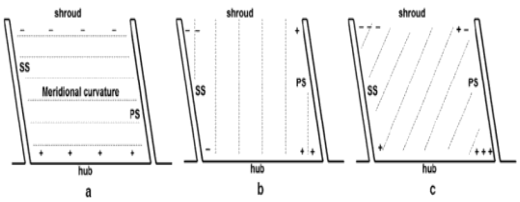
This example concerns single stage centrifugal/ mixed flow configurations. The reason for such a choice is guided by the fact that, for a fairly long period, the research carried out on flows in radial machines was considered too difficult to handle compared with axial machines.

In 1960, Dean and Senoo [44] developed a loss model in the vaneless part downstream of a radial impeller coming from the development of low momentum flows transport along the span-wise meridional path that forms the so-called "jet and wake" structure. It has been first experimentally identified by Eckardt in 1976 [45] and numerically modelled by Moore and Moore [46] in 1980 assuming integral turbulent flow hypothesis. and later in 1993 by Hirsch et al [47-48] through a 3D Navier-Stokes simulation.

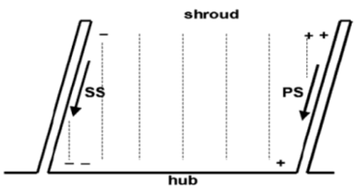
Before these 3D viscous simulations, in the 80's, it has been qualitatively demonstrated that blade leaning may have some effects on the wall boundary layer developments as examined by Van den Braembussche [49] simply performed on a 2D basic inviscid physical analysis as shown in figure 10.



Pressure gradients in a crosswise plane with zero lean. Left: hub to shroud, middle: blade to blade, right: combined resulting effect.



Influence of positive blade lean on pressure distribution



Influence of negative blade lean on pressure distribution

Figure 10. Qualitative effects of different leans on the blade-to-blade pressure gradients.

Around 2010, inverse technique results applied to an industrial mixed flow machine obtained by Goto et al. [50] based on previous Zangeneh's work [51] (see figure 11), quantitively gave what have been initially proposed to reduce the wake at the impeller outlet section.

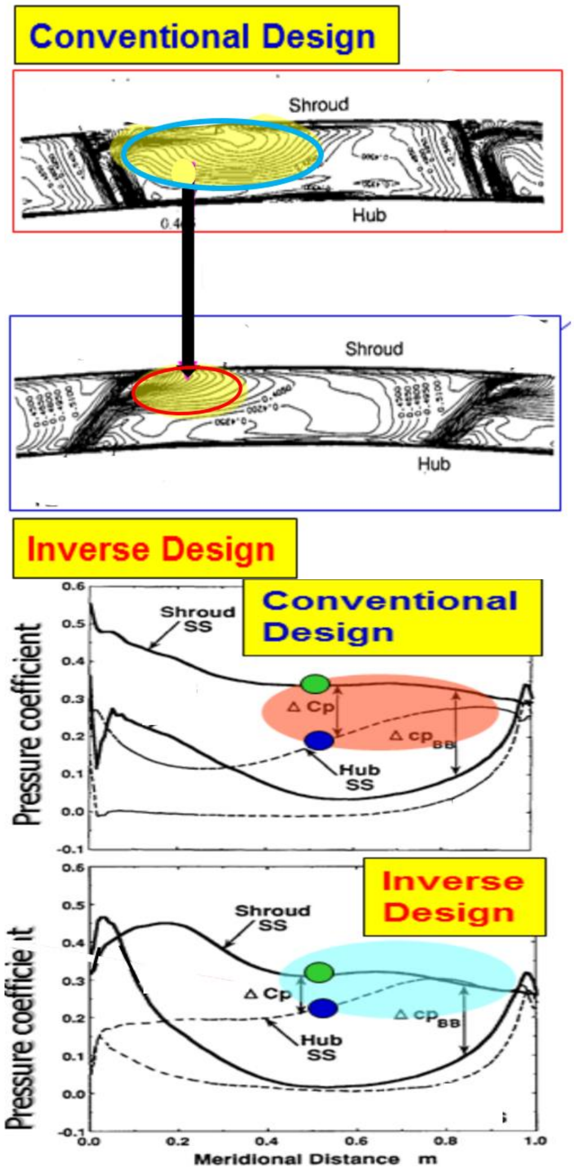
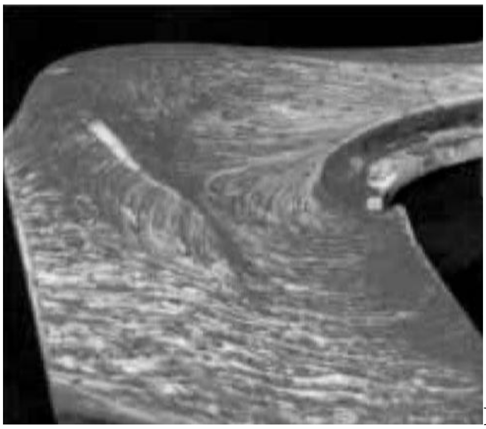


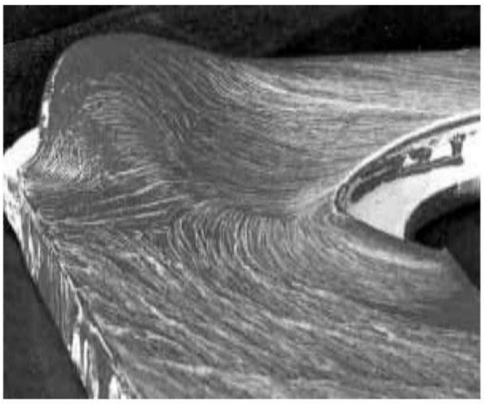
Figure 11. Results of an inverse design approach applied on a mixed-flow impeller pump at design conditions. Adapted from [50].

Recently, in 2019, Merli and Gaetani [52] proposed an extended study including vaneless diffuser performances for 3 different flow rates in a similar geometry including the consequences inside the vaneless diffuser. The study confirmed that negative lean is generally favourable to reduce losses in the complete stage.

5.3. Example 3: Non-axisymmetric turbine end wall profiling

In the early 2000's, end wall contouring attempts have been developed by D G Gregory-Smith et al. [53] in a two-dimensional linear turbine cascade. Spanwise secondary vorticity distribution was successfully improved but not loss coefficient.


P1


P2

The present concept was tested on a 2D cascade, based on inviscid blade-to-blade pressure gradient results. However, when applied to real cases as for a complete stage, overall results were not so good as expected because the rotor placed behind the modified stator was not redesigned. In other configurations, cooling effects coming from the hub platforms completely modify the initial flow inlet conditions entering the end wall region, destroying the initial beneficial effects.

With the implementation of inverse design techniques coupled with cooling injection effects in high pressure turbine stages as shown on figure 13, new end wall designs have been proposed by Burigana et al. [54], with an integration on local purge flow injection in low aspect ratio rotors.

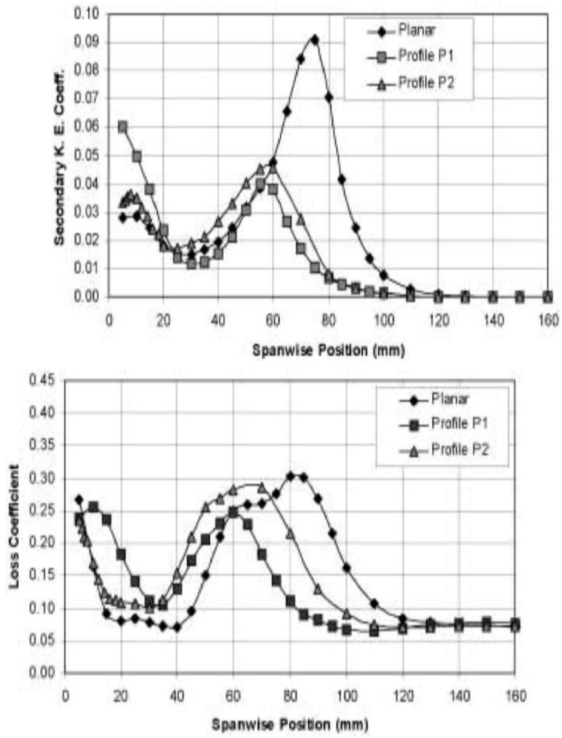


Figure 12. End-wall contouring applied in a turbine cascade. Adapted from [53].

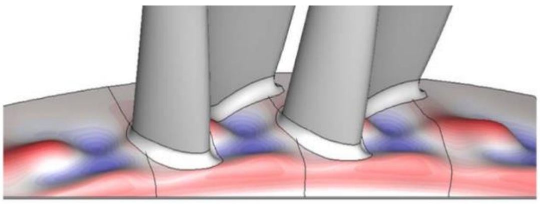


Figure 13. Inverse approach including non-axisymmetric end-wall designs. Adapted from ref. [54].

These last examples, taken for turbine blade rows, show that initial boundary conditions related to flow injection effects strongly modify the end wall design compared with 2D axisymetric inlet boundary layer. This is also the case for axial compressor components as can be seen below.

5.4 Example 4. Inlet leakage effects on performances.

A numerical sensitivity analysis of the hub leakage amount performed by Shabbir et al. [55] shows that induced local deviation angle and loss distribution differences strongly modify the whole spanwise flow distribution especially when the inlet relative Mach number is higher than 1.

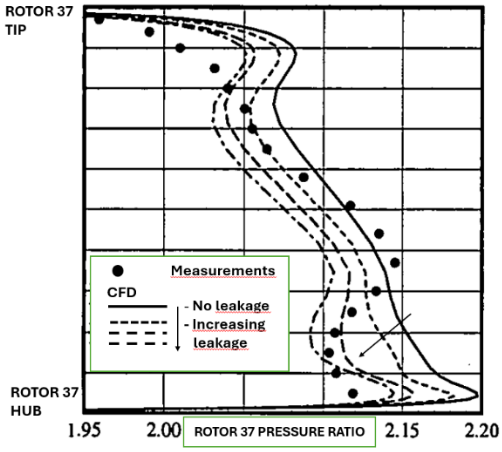


Figure 14. Effect of increasing hub leakage downstream of transonic rotor 37. Hub leakage flow cannot be considered axisymmetric along the blade channel. Adapted from [55].

5.5 Example 5. Cavitation onset and control.

Note: Only pump overall steady performances are here presented without hydraulic turbine cases that must be not forgotten. Unsteady cavitation problems will be discussed in section 7.3

Starting from the 60's up to the 90's, lots of experimental investigations have been carried out in Germany, Switzerland, Italy and France to analyze pumps performance variations under cavitation such as Barrand et al. [56] and Stoffel et al. using noise detection technique [57] among others. Numerical comparisons took place later with experimental comparisons such as Hofmann et al. et al. [58] in a radial pump and by Pelz et al. [59] in which erosion aspects have been introduced.

.

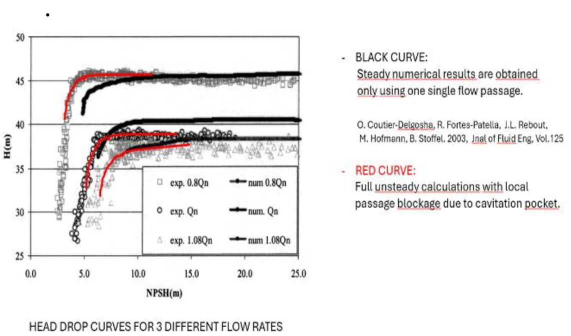


Figure 15. Numerical prediction of the head degradation due to cavitation. Adapted from [58]

Particular attention was also devoted to inducers' designs and performances. They are often used in front of centrifugal pumps to extend working operations when cavitation is present. In this respect, effects on cavitating performances have been also experimentally studied to enlarge single axial inducer performance in the early 2000's by Bakir et al. [60].

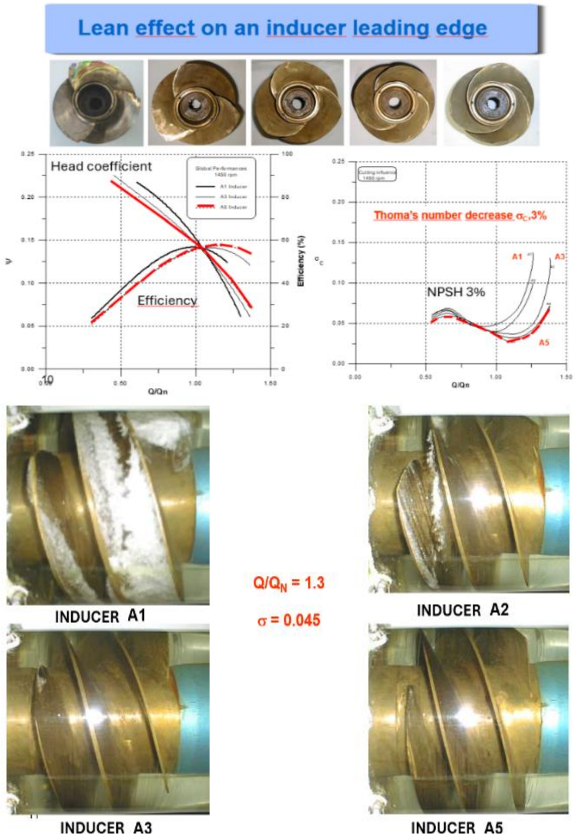


Figure 16. Cavitation visualisation for several inducer designs Adapted from [60].

More recently, an interesting investigation on performances including Inducer coupled with a radial impeller with contra-rotative shafts has been proposed by Dehnavi et al. [61].

These simplified approaches cannot reproduce the whole flow patterns and loss distributions. However, they can explain why quite efficient design improvements have been possible in the past. The implementation of complete 3D viscous softwares allowed to evaluate the relative importance of the missing terms between 2D and 3D inviscid approaches. This will lead to the implementation of multi-objective optimization techniques.

6. MULTIOBJECTIVES OPTIMIZATIONS.

Inverse design methods and optimization techniques brought important Improvements on flow pattern deliveries and, consequently on design improvements. They do help existing basic and acquired knowledge on flow physics, including noise level predictions, exploring large experimental databases and great deal of experience to free oneself from the lack of complete evaluation of several effects coming from technological aspects and other constraints.

However, this served as a basis for setting up a series of adequate design variables to speed up multi-objective optimization procedures including viscous effects that took place along with numerical technique improvements. Figure 17 illustrates how leading-edge modification can improve the cavitation onset in a centrifugal pump (Zhang et al. [62]). Other attempts applied in specific inducers' shape were also proposed like Parikh et al. [63] for example.

At the end, and for validation decision, one has be sure the chosen solution is well adapted to the related application or might be the best one not only for the design point but for the entire flow rate. For example, hydraulic low and medium specific speed machines that are used in land, sea and space applications are supposed to work within large flow rate operations, from shutt-off to high flow coefficients. These constraints lead to a trade-off decision as illustrated in figure 18.

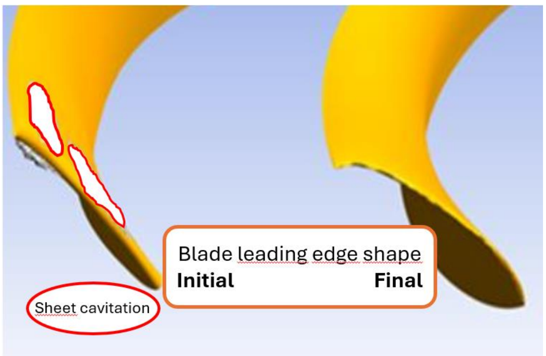


Figure 17. Multi-objectives optimisatio for inlet cavitation control. Adapted from [62].

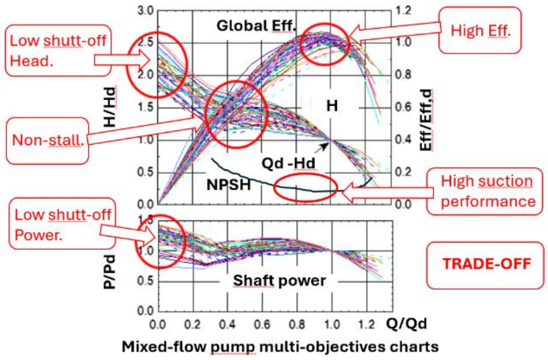


Figure 18. Multi-objectives achievements applied in a mixteflow pump. (Adapted from [50]).

In the same context, the case of high-speed machines has been strongly investigated with more variables due to high Mach numbers, thermal effects and multiple interactions. The choice of the adequate design space to perform efficient muti-objective optimisations becomes a crucial point.

Three separated examples, respectively published in 2001, 2013 and 2024, represent different design parametrisation that have been performed using 3D RANS approaches. The first and last ones are related to blade lean and sweep effects, respectively obtained for a stator blade downstream a high by-pass fan and for a single fan blade can be found in [64] and [65]; the second one concerns end wall and fillet optimisation attempts [66].

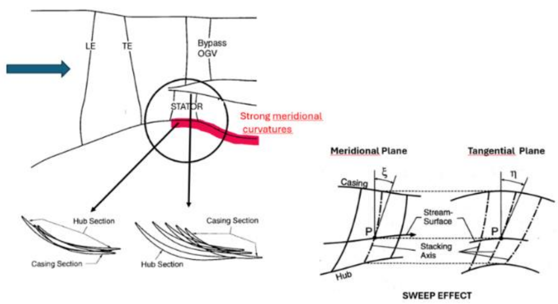


Figure 19. Sweep effects for a stator blade with strong meridional curvatures. Adapted from [64]

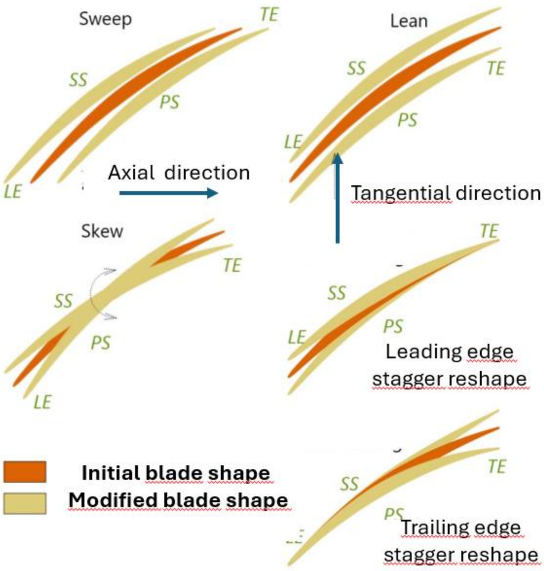


Figure 20. Representation of sweep, lean and skew in a compressor rotor. Adapted from [65]

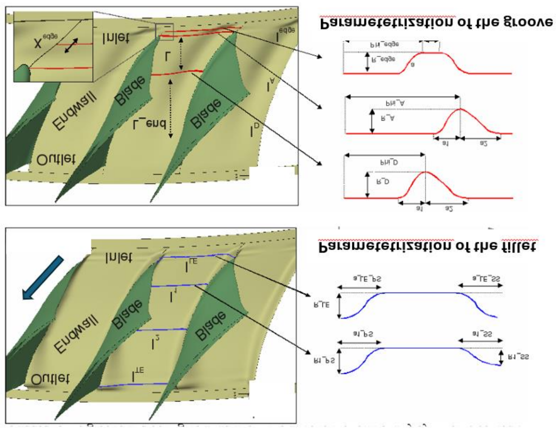


Figure 21. Parametrization of groove and fillet in a compressor blading. Extracted from [66].

More generally, robust designs techniques bring to enforce the following issues:

- check and found possible ultimate performance improvements in a specified design space,
- make possible innovative product development challenge by combining multi-objective and multi-disciplinary design aspects linked to real problems.
- allow to use physical aspects-based design optimization for flow control and systematize design knowledge, which possibly comprehends empirical design practice and /or know-how.
- the innovation design approach should be in close collaboration advance with manufacturing technology innovation. This is because numerically optimized shape can be very complex and difficult to manufacture accurately by conventional or innovative manufacturing technologies. Using fast 3-D printers for example, will expand design space and bring new possibility in product innovation.
- Visualization of multi-dimensional objective space is also important to make efficient trade-off design selection. Continuous challenges can be foreseen, employing numerical optimization, especially by young generations familiar with the digital engineering.

7. MULTI-STAGE UNSTEADY PHENOMENA.

7.1. Unsteady flow classification.

The deterministic and non-deterministic sources identification, listed from Leboeuf [67] (figure 22) and their consequences have been successfully treated when 3D URANS methods start to be used in the early 90’s. They allow to perform more complete flow analysis avoiding the needs for additional models that are necessary to overcome the lost information coming from RANS hypothesis as pointed in the above paragraph.

- *Note: Unsteady sources evaluation also contribute to predict vibration, flutter and aerodynamic noise. For the last item, fast running, approximate analytical approaches and high-fidelity numerical simulations have now reached quite matured tools to meet the needs of the coming decades. These aspects are not treated in the present document. A complete review on turbomachinery noise can be found in the recent paper published by Moreau and Roger in 2024 [68], including key points related to blade passing frequency, vortex shedding and flow separation.*

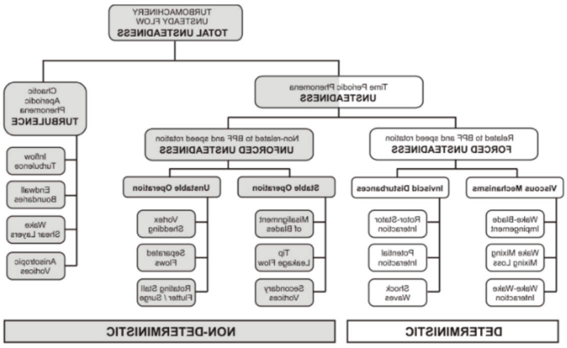


Figure 22. Unsteady flow classification. Extracted from Leboeuf [67].

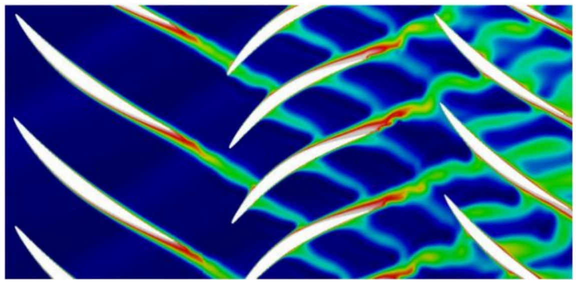


Figure 24. Mid-span wake (visualized from entropy values through fully unsteady calculation) in a 3.5 stage axial flow compressor CREATE. Extracted from [70].

7.2. Forced phenomena.

Gas turbines have numerous stages, low aspect ratio blades with large clearances especially in hot gas stages. Intense secondary flows enhance span-wise transport phenomena. These issues induce different mean flow blockage along the blade span that affects pressure recovery and can produce quite important misunderstanding and wrong data reduction analysis. Taking them at an early-stage design procedure through 3D RANS approach for example, ensure a correct evaluation and a good interstage matching through controlled circumferential flow averaging techniques especially for high-speed machines as already pointed just before.

Before 2000's, URANS methods were generally used both for in single stage configuration to evaluate the lost information coming from through flow methods. An example is given in Figure 23 about entropy contour differences using 2D steady approach and 3D unsteady one in a turbine stage [69]. When fully 3D URANS is performed as for the case of a multistage compressor, presented on Figure 24 which comes from Courtiade's PhD report [70].

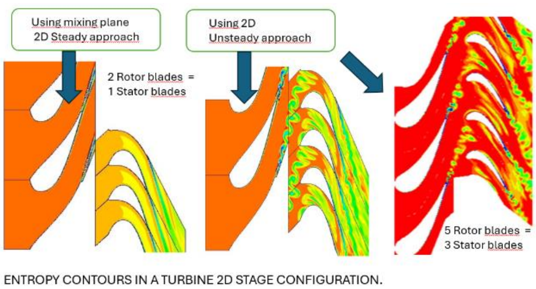


Figure 23. Differences on entropy contours resulting from steady and unsteady assumption. Adapted from [69].

The related major fluid mechanics issues that can be extracted from full 3D URANS results can be listed as follow:

a) Effects of spanwise and circumferential transport of wakes.

Three main phenomena are responsible for wakes' decay:

- Viscous mixing, which reduces the velocity deficit of the wakes regarding to the free stream. This is associated with dissipation due to the wake stretching when traversing the downstream rotating and/or fixed rows. In 1990, Poensgen and Gallus [71]. had already noticed that the presence of a downstream stator induced a decay of the wakes of a rotor twice faster than an isolated rotor configuration
- Circumferential redistribution involving interactions of incoming wakes on transitional or turbulent blade BL.
- Wake straining and wake recovery process due to blade passage transport. Different consequences for compressor (decelerating mean flow) and turbine (accelerating mean flow) cases have been explained and modelled.

The velocity deficit of the wake in comparison with the free stream, also called "negative jet". This produces a fluid accumulation on the pressure sides of the blades, as visible on Figure 25, extracted from Mailach *et al.* (2008, [72]). The velocity triangle at rotor inlet shows indeed that the local velocity deficit is responsible for the accumulation of the wakes on the pressure sides. Therefore, the wakes get locally thinner in the blade passages near suction side and thicker near pressure side which thickens the boundary layers on the blade pressure side. This is reinforced at tip sections when tip leakage flows are present.

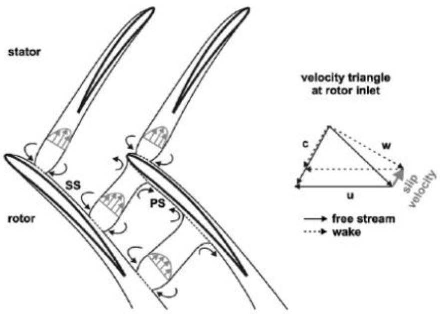


Figure 25a. Wake transport inside a rotor channel without tip leakage.Adapted from [72].

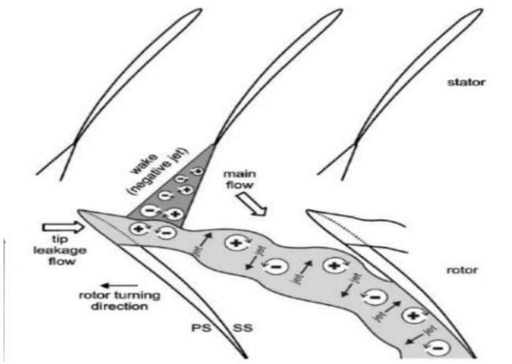


Figure 25b. Wake transport inside a rotor channel with tip leakage.Adapted from [72].

b) Misinterpretation of experimental results

- After a rotor, depending on the location of the measuring plane, interpretation of results may be confusing due to time and/or mass averages techniques and unsteady effects.

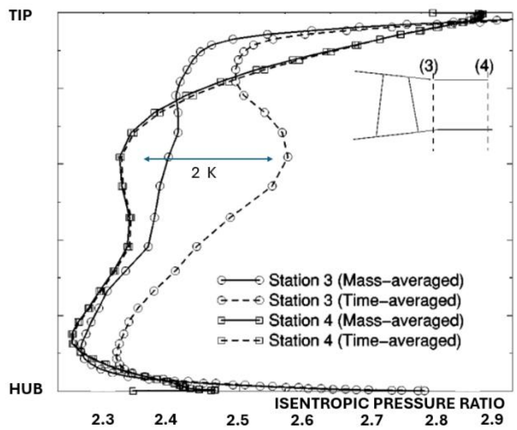


Figure 26. Axial evolution of the hub to shroud radial total temperature profile identified by Adamczyk [30].

c) Spanwise redistribution of total temperature and momentum that may produce hot spot temperature impacting turbine blade life.

d) Detection and identification of several modes that can be compared with fast response wall pressure transducers explained in the works performed by Ernst et al. [73] and Courtiade and Ottavy [74], both in 2011.

e) Unforced phenomena simulations as detailed below.

7.3. Unforced phenomena.

Throughout the last 30 years, several sources of unforced phenomena have been identified mainly from experimental investigations, long before the use of unsteady numerical tools.

Some particular phenomena are listed below:

- Wake instabilities as discussed before
- Vortex shedding. Vortex shedding effects on pressure loss (see Roberts and Denton [75]) and fluctuations behind turbine blades (see Cicatelli and Sieverding [76]).
- Aerodynamic interactions between core flow and shroud cavity as explored by Tanga et al. [77]. On figure 28, a clear peak at the blade passing frequency is detected, but the spectrum is dominated by several other peaks over a range at low frequencies between 0.35 and 0.8 times *BPFs*. Thus, the passing of the blades is not the only source of oscillations.
- Other coupling effects like blade flutter, shock instability, rotating stall and surge.

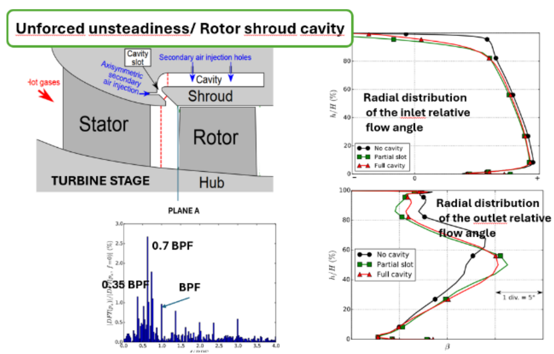


Figure 27. Unforced phenomena associated with a rotor should cavity in an axial turbine stage [77].

- Corner separation and corner stall. These flow features were studied by Schulz et al. in the 90's ([78], [79], [83]), followed by the attempt of Lei et al. [81], who propose an interesting simple 1D criteria for its onset.

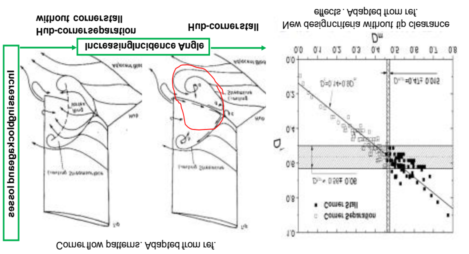


Figure 28. Flow topology of the corner separation and corner stall and the aerodynamic performance for the end wall corner flows. Adapted from [78] (Left) and from [81](Right).

7.3.1. Rotating stall.

In most cases, at low flow rates, the axisymmetric initial flow becomes unstable (Note that a non-axisymmetric flow can be highly stable). In the circumferential direction, the flow divides into regions of low flow rate called "stall cell patterns" and un-stalled regions in which the flow rate is larger than the mean flow rate just before critical stall point of operation.

This point has been an active research field, both in axial and radial machines, aiming to understand and predict its onset then try to control its effects.

Rotating stall onset arises in different ways in radial and axial machines and this is mainly due to different performance characteristics curves and different machine length-volume and mean velocity flow scales (time scales)

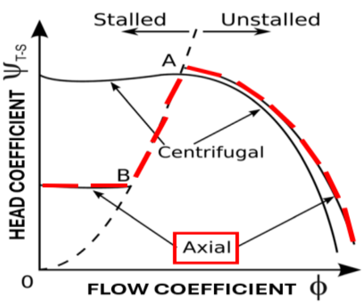


Figure 29. Typical curves of head coefficient versus flow coefficient in axial and centrifugal compressors.

a) Vaneless diffuser rotating stall.

This phenomenon was identified in the 60's by Jansen [83]. Many other experimental and numerical investigations were performed mainly after 1990, thanks to advanced experimental techniques like PIV and numerical simulation both in 2D and 3D approaches such as Fring and Van den Braembbusche [84](1985), Moore [85] (1989), Ferrara et al. [86] (2004) , Ljevar et al. [87] (2006) , Dazin et al. [88] [89] (2008 and 2011).

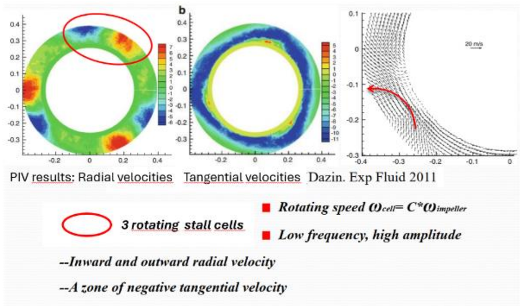


Figure 30. Rotating stall cells identification in a vaneless diffuser from PIV results. Adapted from [89].

b) Axial flow compressors.

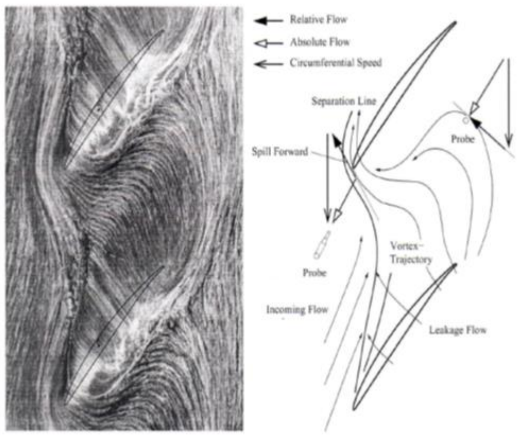


Figure 31. Secondary flow visualization in a 2D compressor cascade with tip leakage. Adapted from [9].

In axial flow machinery, it is generally recognised that transition to axial compressor rotating stall can develop in two different ways, the so-called spike type and modal type route to rotating stall. Figure 31 shows the visualisation of limiting streamlines that occur at high incidence angles in a 2D compressor rotor cascade including tip-leakage (from [9]).

An exhaustive review paper has been published by Day. [90] on this topic where only aerodynamic sources are present.

In some cases, acoustic resonance can trigger rotating stall or surge as detected by Hellmich and Seume [91].

More research insights are in progress about the combined effects of incidence, tip leakage and low momentum radial migration along the blades during the rotating stall route. Stall precursor detection for flow control activation is still an important topic to solve.

Recent works (Romano et al. [92]) looking at distinct type routes due to passive and active flow control and inlet distortions, brought new insights for better understanding on such phenomena.

In case of forced inlet inhomogeneities, an analysis of unsteady pressures measured at the casing shows that in the baseline case (with no injector and no grid), the transition to stall is of spike-type whereas, if the intensity of the inhomogeneity is increased, the stall can be of modal type or a mix between spike and modal.

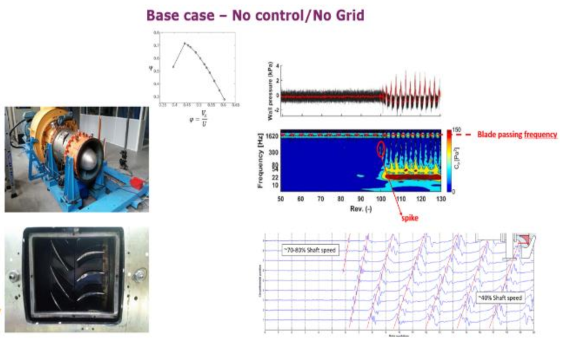


Figure 32. Time frequency charts in an axial flow compressor stage. Rotating stall inception for uniform axi-symetric inlet conditions.

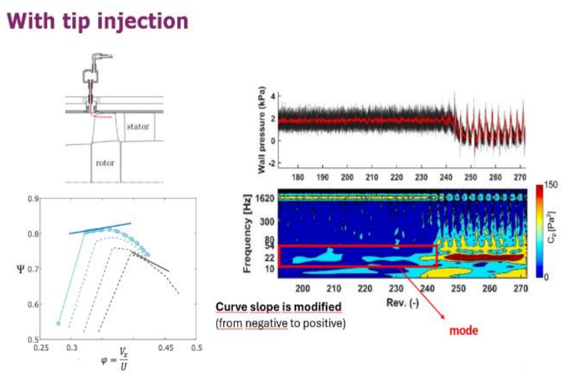


Figure 33. Time frequency charts in an axial flow compressor stage. Rotating stall inception for uniform axi-symetric inlet conditions with tip injection.

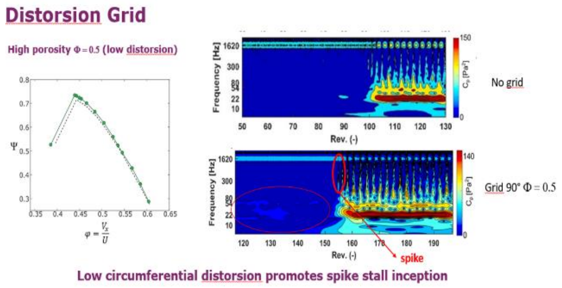


Figure 34. Time frequency charts in an axial flow compressor stage. Rotating stall inception for non-uniform inlet conditions.

7.3.2. Cavitation

Cavitation occurrence not only influences overall pump performances like already seen in section 5 (example 5) but can also produce unsteady phenomena especially in hydrofoils and axial pump inducers' parts.

Their designs are strongly related to the application since strong rotating pressure fluctuations at frequencies that can be coupled with the whole system (in the draft tube of Francis hydraulic turbine or POGO effects in cryogenic pump rocket propulsion systems for which an example is given on Figure 35a and 35b.

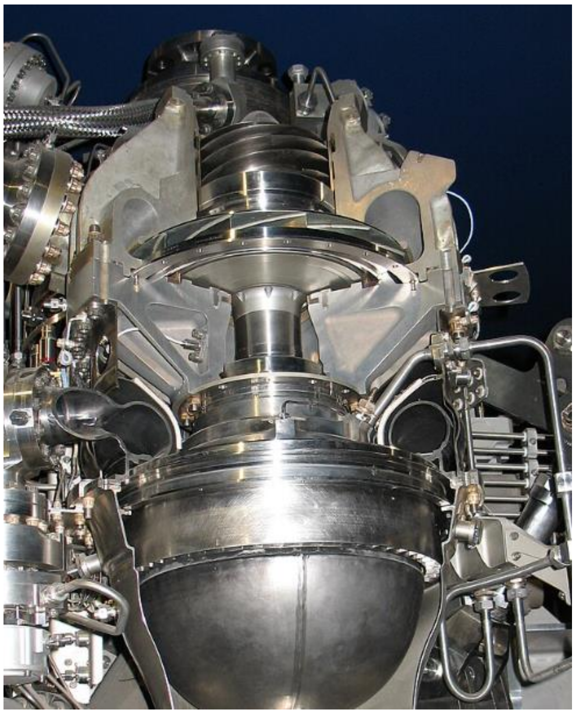


Figure 35a. Cryogenic pump for space application.

Figure 35b. New compact pump design.

It is so mandatory to improve existing flow unsteady models to better capture the re-entrant jet description which is strongly related to the local two-phase mixture density distribution. As pointed out in the published works referred in [93] to [97], it is responsible for unsteady pressure distribution along the blade chord related to the Strouhal number linked to the bubble detachment sequences that are transported in the main flow.

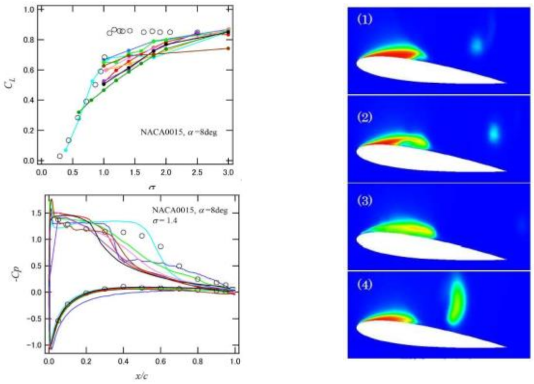


Figure 36. Blade pressure distribution with sheet cavitation (left). Unsteady sheet cavitation on the suction surface of a blade profile (right).

8. TOWARDS ROBUST DESIGN TOOLS. MACHINE LEARNING (ML).

During the last 20 years, more and more knowledge and strong experience has been acquired from basic experimental analysis to model derivations and numerical simulation packages. They have had strong consequences for better flow descriptions and design integrations and lead to important efforts to implement a robust design concept.

Robust design concept is based on good evaluation and calibration of performances for the whole range of operation whatever the accuracy of the model is, if it has been well calibrated according to the desired objectives. This should be inserted at an early stage in the design procedures

For this purpose, the need for detailed and well documented information on realistic test rigs is more and more recommended and even mandatory. There are still many existing experimental results that are available but with a lack of detailed information about inlet turbulent scales or inlet boundary layers for example.

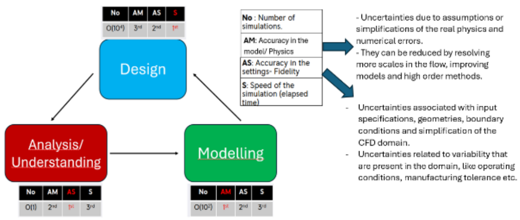


Figure 37. Design paradigm involving accuracy and simulation speed [98].

Robust design mainly depends on CFD overall accuracy that can be considered as a function of models and settings accuracy as shown on figure 37 extracted from Vasquez and Adami [98].

Reducing the sources of inaccuracies require precise information on:

1. Detailed geometry description: non-axisymmetric casings and blades, blade settings, hubs and shroud cavities and gaps, rotor tip clearance, stator hub clearance, geometry variations, buttons, blade fillet and surface finish, manufacturing tolerances,
2. Numerical errors due to mesh and/or finite difference approximations

3. Settings: geometry fidelity, real geometry discrepancy, impact of manufacturing variability on performances.

4. Models:
- trusting CFD's physics and mathematics according to the level of approximation,
- evaluate the aleatory uncertainty due to boundary conditions,
- inserting correct inflow and outflow boundary conditions: all leakage flows, bleed flow, purge and cooling flows (mainly for turbines)
- Steady flow assumption consequences.

Below, one can find 3 different examples that can illustrate different ways to evaluate existing empirical laws and built robust design tools depending on the level of approximation with an adequate design space choice. This corresponds to the so-called "machine learning" techniques that speed up prediction times, analyse and manage uncertainty and reconcile simulations with available data with better prediction accuracy. "Such techniques facilitate faster and more robust searches of the design space, with or without the help of optimization methods" as pointed out by the review paper proposed by Hammond et al. [99].

8.1. Example 1.

Even if it may seem paradoxical, it turns out that for some cases, a resumption of the analysis of isolated phenomena can enrich robust design approaches. The following research works from Raina et al. [100] [101] give some answers the two following questions:
- how to properly extract individual loss levels when strong interactions occur between vortices and wakes?
- to what extend is it possible to use independent loss models' assumption to predict overall losses?

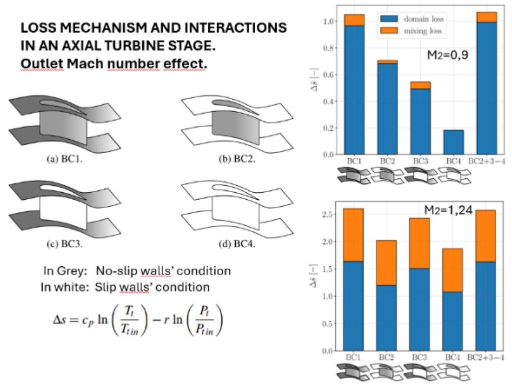


Figure 38. Evaluation of loss mechanism in an axial turbine stage. Outlet Mach number effect. Adapted from [100].

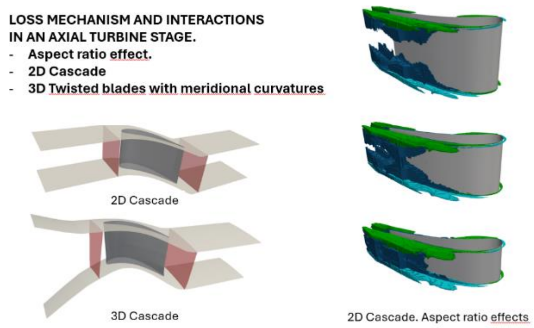


Figure 39. Evaluation of loss mechanism in an axial turbine stage. Aspect ratio effect. Adapted from [100].

8.2. Example 2.

How to use the differences between low and high-fidelity numerical simulations for a 2D case, under low Reynolds number like for high altitude flight conditions (see Bergmann et al. [102]).
- Uncertainties issued from RANS simulations do not produce constant offset of results compared with LES ones. They are flow dependant especially for off-design conditions.
- High fidelity scale-resolving simulations at low Reynolds number offer powerful tools for early design prediction processing at challenging operating points
- Evaluation and analysis on the differences between RANS and LES can enhance design reliability.

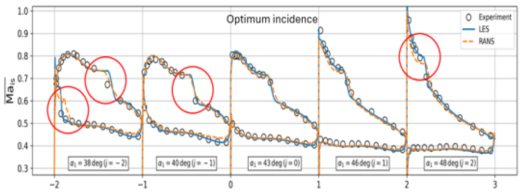


Figure 40. Comparison between RANS and LES in a compressor blade at low inlet Reynolds number condition. Adapted from [102].

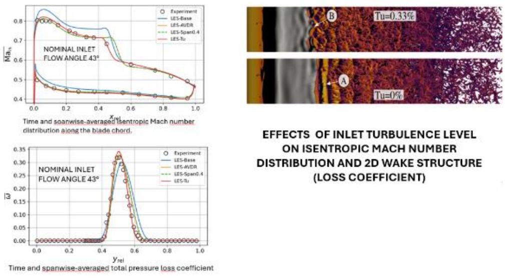


Figure 41. LES results in a compressor blade at low inlet Reynolds number condition. Effect of the inlet turbulence level. Adapted from [102].

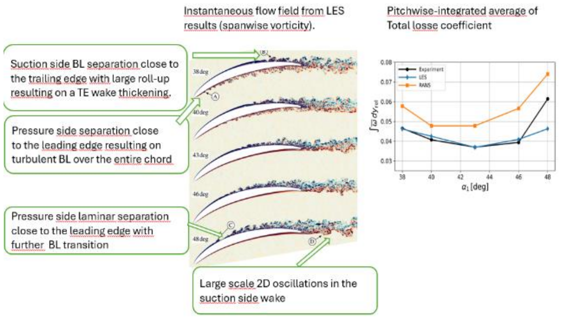


Figure 42. Comparison of total loss coefficient with incidence angle between RANS and LES in a compressor blade at low inlet Reynolds number condition. Adapted from [102].

8.3. Example 3.

What are the necessary conditions for the detection of pump sump submerged vortex onset? (Kato [103]).

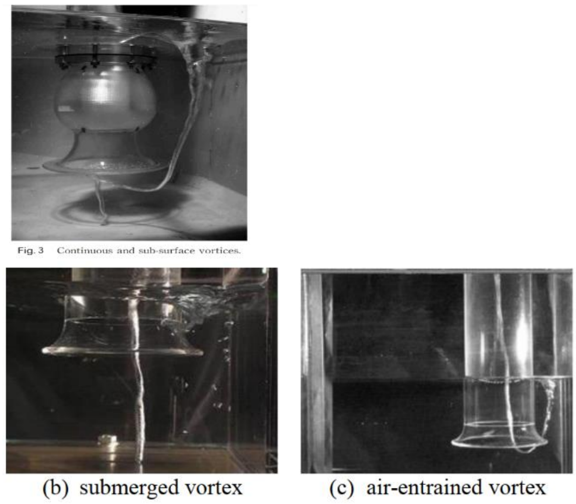


Figure 43. Experimental observation of submerged and air-entrained vortices.

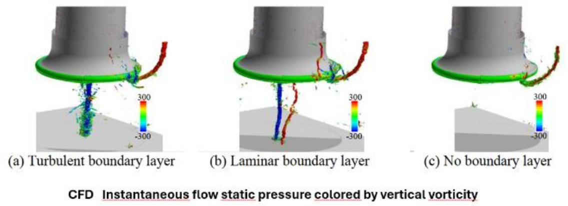


Figure 44. CFD LES results for vortex detection simulations in a pump sump. Adapted from [103].

One can observe that, in recent years, scale resolving simulations such as Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS), as opposed to time-averaging methods like (U)RANS, are more frequently used for design aid. Figure 45 shows where each method lies on the trade-off between accuracy and computational expense and the correlation of these methods to the energy spectrum. Despite their undeniable superiority to (U)RANS, LES with appropriate grid resolution is still computationally prohibitive, while it is more common for the wall-bounded and statistically steady flows. Scale resolving simulations offer superior resolution allowing designers to determine not only *what* is happening in the fluid flow, but also *why* it is happening. They produce accurate numerical datasets to benchmark and improve (U)RANS models.

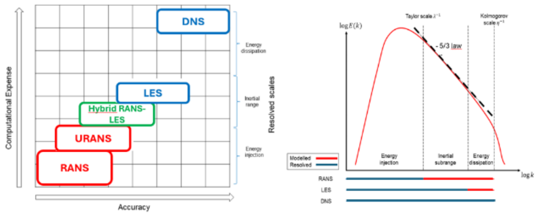


Figure 45. Computational expense against accuracy depending on the approach (left) and capability of each method to model or resolve energy spectrum (right).

9. ARTIFICIAL INTELLIGENGE (AI).

Artificial intelligence is asserting itself as a powerful tool for amplifying human decision-making capabilities and overcome difficult to solve problems in all aerodynamic fields.

Presently, the application of AI in turbomachinery field is at an early stage of exploration We are still far from fully mastering the extent and consequences of confrontation and mastery of this new paradigm.

Many investigations have still to be explored in several areas like real 3D inverse design, experimental testing, intelligent control systems and maintenance.

Take advantage of previous works and models to build improved and reliable rules extracted from experiments and CFD. This needs important background and clear knowledge of the relative importance of each isolated phenomena and interactions.

For any kind of turbomachinery applications, AI can be integrated into any step of the complete conception procedure, starting from the design step up to the validation and the maintenance.

AI can be employed for autonomous decision-making capability in aerodynamic optimization procedure and active flow control of turbomachines, generating optimal aerodynamic solutions and complex control strategies that surpass human capabilities.

AI uses machine learning data to perform most aerodynamic tasks whatever the level of approximation is.

An example of specific work devoted to centrifugal compressor families using AI-based fast design method can be found in [104].

It can be considered as an upgrade of up-to-date aerodynamic design systems allowing to replace the need for additional empirical expertise and numerical results.

It must allow to gather and share several kinds of data and information between the different fields of specialization, enforcing interactive 3D design optimization and real-time prognosis including in-situ control and global maintenance analysis.

For numerical simulations, AI can help to extracts knowledge from high-fidelity data in DNS simulations and experimental measurements to improve transition models for RANS and URANS simulations and consequently improve the simulation accuracy.

For experimental tests, AI can help to clean, complete, reconstitute or restore missing or incomplete or limited measuring data sets by incorporating physical constraints. AI could help to reconstruct overall flow fields from limited measurement data by physics-informed neural networks.

When validation is required, AI only acts as a supplement to classical methods that use numerical simulations and/or experimental tests.

The question is: what should be the researcher's involvement to produce results for AI data base mining?

10. CONCLUSIONS.

Whatever the evolution of future needs related to energy transition, drastic and rapid changes in demand are expected in many sectors, including the one for turbomachinery applications. In this context, new aerodynamic design processes exploiting the existing know-how and enriched by efficient multi-disciplinary techniques could quickly react to demand variations and ensure technical innovation and end-users' success.

Many of CFD approached are now mature and well advanced even if further improvements can be expected in near future as for laminar-transition evaluation accuracy and turbulence modelling. For such problems, machine learning have already a specific impact on CFD within the space of turbomachines specifically, where data from higher-fidelity simulations are used to train models issued from lower fidelity calculations.

Additionally, CFD assessments need extensive well documented experimental investigations and data base elaboration for a large multidisciplinary design space and in realistic geometries including all physical aspects including noise levels predictions

.

- For high-speed machines, there is still research needs to analyse 3D shock-structures interactions, coupled with deformation, blade flutter, vibration modes detection, off-design aspects including stall and surge, fast transient phenomena for safety problems in adiabatic and diabatic conditions, purge and leakage flow interactions with thermal effects, fouling and cleaning assessments. All these different aspects that involve different engineering fields must be integrated to produce new architectures like the ones shown in figure 46 for air-breathing engines.

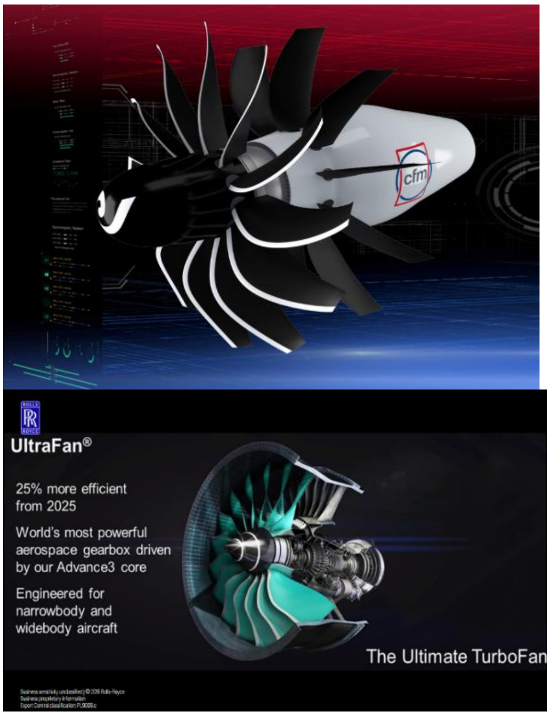


Figure 46. CFM-RISE Program (mid 2030). RR UltraFan (for narrow and wide body aircrafts)

- For hydraulic machines, extended estimation for high fluid viscosity and non-Newtonian behaviour should be foreseen, pump as turbine configurations should be more explored including better predictions on multiphase flow and cavitation instabilities.

A recent example of the open test case CATANA for a high-speed fan [105], provided by the Fluid Mechanics and Acoustic laboratory in Lyon illustrates what could be done for the whole turbomachinery community.

However, many flow and heat transfer processes cannot be easily tested. CFD provides the ability to theoretically simulate any physical conditions. It allows great control over the physical process and provides the ability to isolate specific phenomena that can be also tested experimentally for comparison and validation.

It may happen that, for some cases, initial and boundary conditions may be sufficient to run a computational simulation but does not guarantee agreement with measured data and design experience. Recent developments based on Physics-Informed Neural Networks (PINNs) offer a very interesting potential to combine all sources of available information in a single constrained optimisation network. Measured data, data from high-fidelity sources, theoretical physical laws and tried and tested empirical relationships can all be considered simultaneously by incorporating them in the loss function of a learning machine.

Today, producing a very good design for a turbomachinery component is not any more the main target. Compromise must be checked due to increasing technical constraints. For example, evolutions to new architectures, simplified geometries, compact designs, fast machining, long life targets, gain on weight and time of repairs, life duration, modelling of other fluids, new cycles, electric powering, energy distribution for safe operations and so on…. This will change the future to partially or completely solve each related task facing numerous multidisciplinary interactions.

Past knowledge cannot be ignored and must be stored for machine learning capability and efficient data base elaboration. In this respect, it is very important that a designer using CFD appreciates the underlying assumptions and limitations of the numerical techniques that are available to him. It is consequently very important to present all the predictions coming from future advanced developed models with uncertainty evaluations helping engineers for confidence in the results obtained and justify the design decisions taken as a result.

ACKNOWLEDGEMENTS

The author thanks the French ATTAG (Association Technique des Turbines à Gaz) for its support in collecting part of the information presented in this document.

REFERENCES

[1] Cumpsty N.A, 2010, "Some lessons learned". ASME Jnal of Turbomachinery, vol. 132, October 2010.

[2] Hawthorne W.R., 1955, Rotational flow through cascades, Quart. J. Mech. Appl. Math., 8, 3, 266-279

[3] Horlock, B. Lakshminarayana and J. H. 1963. "Review : Secondary Flows and Losses in Cascades and axial - flow turbomachines." Int. J.Mech.Sci. 5: 287–307.

[4] Horlock J.H., Lakshminarayana B., 1973, Secondary flows: theory, experiment, and application in turbomachinery aerodynamics, Ann. Rev. Fluid Mechanics, 5, 247-279 17.

[5] Lakshminarayana B., Horlock J.H., 1973, Generalised expressions for secondary vorticity

using intrinsic co-ordinates, J. Fluid Mech., 59, Part 1, 97-115

[6] Marchal Ph., Sieverding C.H., 1977, Secondary flows within turbomachinery bladings, Proc. AGARD-CP-24.

[7] Papailiou. K., Flot. R., Mathieu. J. Secondary flows in Compressor Bladings. ASME J. Eng. Power. Apr 1977, 99(2): 211-224 (14 pages). https://doi.org/10.1115/1.3446274

[8] Gallus, H.E. and Kummel, W. 1977. Secondary Flows and Annulus Wall Boundary Layers in Axial-Flow Compressor and Turbine Stages. AGARD CPP No. 214. Paper No. 4: 1–15.

[9] Langston, L.S. 1980. Crossflows in a Turbine Cascade Passage. *ASME Journal of Engineering for Power*. **102**: 866–874

[10] Sieverding, C.H. 1985. Recent Progress in the Understanding of Basic Aspects of Secondary Flows in Turbine Blade Passages. *ASME Jour. of Turbomachinery*. **107**: 248–257.

[11] Sharma, O.P., and Butler, T.L. 1987. Predictions of Endwall Losses and Secondary Flows in Axial Flow Turbine Cascades. *ASME Jour. of Turbomachinery*. 109: 229–236.

[12] Hodson H.P., Dominy R.G., 1987, Three-dimensional flow in a low pressure turbine cascade at its design conditions, Trans. ASME, J. Turbomachinery, 109, 278-286

[13] Schulz, Gallus, Laksminarayana. Three-Dimensional Separated Flow Field in the Endwall Region of an Annular Compressor Cascade in the Presence of Rotor-Stator Interaction Journal of Turbomachinery OCTOBER 1990, Vol. 112, pp 669- 678)

[14] Kang S 1993 *Investigation on the Three-Dimensional Flow Within a Compressor Cascade with and without Tip Clearance* PhD Thesis, Dept. of Fluid Mech., Vrije Universiteit Brussels.

[15] Langston, L.S. 2006. "Secondary Flows in Axial Turbines-A Review." Annals of the New York Academy of Sciences.

[16] Wu, C. H.,1952, A General Theory of Three-Dimensional Flow in Subsonic or Supersonic Turbomachines of Axial, Radial and Mixed Flow Type, NACA TN 2604.

[17] Smith, L., H., Jr., 1969, Casing Boundary Layers in Multistage Compressors, Proc. Symp. Flow Research on Blading.

[18] Kerrebrock, J. L., and Mikolajczak, A. A., 1970, Intra-Stator Transport Of Rotor Wakes abd Its Effect on Compressor Performance, ASME Journal. Eng. Power, Vol. 92,p. 359
.

[19] Bosman C, and Marsh H., 1974, "An Improved Method for Calculating the Flow in Turbomachines, Including a Consistent Loss Model," *Journal of Mechanical Engineering Science*, Vol. 16, No. 1, pp. 25–31.

[20] Hirsch C., and Warzee , G., 1976, "A Finite-Element Method for Through-Flow Calculations in Turbomachines," *ASME Journal of Fluids Engineering*, Vol. 98, pp. 403–421
.

[21] Denton J. D., 1978, "Throughflow Calculations for Transonic Axial Flow Turbines," *ASME Journal of Engineering for Power*, Vol. 100, pp. 212–218.

[22] Denton, J. D., and Singh, U., K., 1979, Time Marching Methods for Turbomachinery Flows Calculations, VKI-Lecture Series, No . 7. VKI

[23] Adamczyk, J. J., 1985, "Model Equation for Simulating Flows in Multistage Turbomachinery," ASME Paper No. 85-GT-226.

[24] Jennions, I.K., and Stow, P., 1986, The importance of Circumferential Non-uniformities in a Passage-Averaged Quasi-Three-Dimensional Turbomachinery Design System, ASME Jnal Eng. Gas Turbine Power, Vol. 108, pp. 240-245.

[25] Dawes, W. N.,1990, Towards Improved Throughflow Capability: The Use of 3D Viscous Flow Solvers in a Multistage Environment, Paper 90-GT-18. Gas Turbine and Aeroengine Congress and Exposition—June 11-14, 1990—Brussels, Belgium**.**

[26] Shabbir, A., Celestina, M. L., Adamczyk, J. J., and Strazisar, A.,1997, The Effect of Hub Leakage Flow on Twi High Speed Axial Flow Compressor Rotors, ASME Paper No. 97-GT-346

[27] Wellborn, S. R., and Okiishi, T.H., 1998, The Influence of Shrouded Stator Cavity Flows on Multistage Compressor Performance, ASME Jnal of Turbomachinery, Vol. 121, pp. 486-497
.

[28] Escuret, J. F., and Veyssiere, P. H., Villain, M., Savarese, S., Bois, G., and Naviere, H., 1997, Effect of a Mismatch Between the Buttons of Variable Stator Vanes and the Flowpath in a Highly Loaded Transonic Compressor Stage, ASME Paper No. 97-GT-471.

[29] Adkins, Jr, G. G., Smith, Jr, L.H ,1982, Spanwise mixing in axial machines, Journal of Engineering and Power, Jan 1982, 104(1): 97-110 (14 pages). https://oi.org/10.1115/1.3227271

[30] Adamczyk, J. J., Aerodynamic Analysis of Multistage Turbomachinery Flows in Support of Aerodynamic Design, 1999, International Gas Turbine Institute Scholar Lecture, Journal nal of Turbomachinery, 2000, Vol. 122

[31] Denton, J. D. Some limitations of Turbomachinery CFD, Proceedings of the ASME Turbo Expo 2010: Power for Land, Sea and Air June 14-18, 2010, Paper GT2010-22540 Glasgow, UK.

[32] Pinckney, S. Optimized turning-vane design for an intake elbow of an axial-flow compressor. NASA Report TN D-3083, 1965.

[33] Neumann, B., The interaction between geometry and performance of a centrifugal pump. Mechanical Engineering Publications Ltd., London.

[34] Flathers, M., Bache, G., Rainsberger, R.: An experimental and computational investigation of flow in a radial inlet of an industrial pipeline centrifugal compressor. ASME Journal of Turbomachinery, Vol. 118, No 2, 1996, pp. 371-377

[35] Ayder, E., Van den Braembussche, R.A.: "Experimental Study of the Swirling Flow in the Internal Volute of a Centrifugal Compressor", ASME Paper 91-GT-7.

[36] Ayder, E., Van den Braembussche, R.A.: "Experimental and Theoretical Analysis of the Flow in a Centrifugal Compressor Volute", Trans. ASME, Journal of Turbomachinery, Vol 115, No. 3, July 1993, pp. 582-589.

[37] Ayder, E., Van den Braembussche, R.A.: "Numerical Analysis of the Three-Dimensional Swirling Flow in Centrifugal Compressor Volutes", Trans. ASME, Journal of Turbomachinery, Vol. 116, Nr. 3, July 1994, pp. 462-468

[38] Van den Braembussche**,** R. A., Ayder, E., Hagelstein, D., Rautenberg, M., Keiper, R., Improved model for the design and analysis of centrifugal compressor volutes, Proceedings of the 1998 International Gas Turbine & Aeroengine Congress & Exhibition - Stockholm, Sweden

[39] Van den Braembussche, R.A. [6] ((2006) Flow and Loss Mechanisms in Volutes of Centrifugal Pumps. In Design and Analysis of High Speed Pumps (pp. 12-1 – 12-26). Educational Notes RTO-EN-AVT-143, Paper 12. http://www.rto.nato.int/abstracts.asp.)

[40] Zangeneh, M., 1991, "A Compressible Three Dimensional Blade Design Method for Radial and Mixed Flow Turbo-machinery Blades," Int. J. Numerical Methods in Fluids, Vol. 13, pp. 599-624.

[41] Zangeneh M (1994) Inviscid-viscous interaction method for three-dimensional inverse design of centrifugal impellers. J Turbomach 116(2):280–290. https://doi.org/10.1115/1.2928362

[42] Zangeneh, M., Daneshkhah, K.2009, A Fast 3D Inverse Design Based Multi-Objective Optimization Strategy for Design of Pumps . Conference paper FEDSM, DOI:10.1115/FEDSM2009-78443

[43] Demeulenaere, A. and Van den Braembussche, R**.** Three-dimensional inverse design method for turbomachinery blading design. ASME paper 96-GT-39, 1996

[44] Dean, R.C., Jr.; Senoo, Y. Rotating Wakes in Vaneless Diffusers. *J. Basic Eng.* **1960**, *82*, 563–570

[45] Eckardt, D. Detailed Flow Investigations within a High-Speed Centrifugal Compressor Impeller. *J. Fluids Eng.* **1976**, *98*, 390–399

[46] Moore, J., Moore, J.G., 1983a. "Entropy production rates from viscous flow calculations. Part i A turbulent boundary layer flow," In: Proc. ASME International Gas Turbine Conference and Expo 1983 83-GT-70. Phoenix, AZ, USA.

[47] Hirsch Ch, Kang S and Pointet G 1996 *A Numerically Supported Investigation on the 3D Flow in Centrifugal Impellers. Part I: The Validation Base* ASME Paper 96-GT-151

[48] Hirsch Ch, Kang S and Pointet G 1996 *A Numerically Supported Investigation on the 3D Flow in Centrifugal Impellers. Part II: Secondary Flow Structure* ASME Paper 96-GT-152.

[49] Van den Braembussche, R., 1985, *Description of Secondary Flow in Radial Flow Machines* Published in Thermodynamics and Fluid Mechanics of Turbomachinery Volume II, Edited by Ucer A S, Stow P and Hirsch Ch, NATO ASI Series, pp. 665-684

[50] Agira GOTO, IJFMS Dec 2015, Historical perspective on fluid machinery flow Optimisation in an industry. Vol9, No 1, Jan- March 2016

[51] Zangeneh, M., Goto, A., Harada, H., [8] , 1998, "On the Design Criteria for Suppression of

Secondary Flows in Centrifugal and Mixed-Flow Impellers," ASME Journal of Turbomachinery, Vol.120, pp723-735

[52] Merli, F., and Gaetani. P., *NUMERICAL* INVESTIGATION OF THE LEAN EFFECTS IN CENTRIFUGAL COMPRESSORS, Proceedings of 13th European Conference on Turbomachinery Fluid dynamics & Thermodynamics ETC13, April 8-12, 2018; Paper ID: ETC2019-143 Lausanne, Switzerland

[53] Gregory-Smith, D.G., Jayaraman, G.P., Harvey, N. W. and Rose, M. G. End wall contouring Gregory Smith Secondary flow control. Proceedings of the Institution of Mechanical Engineers Part A Journal of Power and Energy · September 2001, DOI: 10.1243/0957650011539020

[54] Burigana. M., Verstraete. T., Lavagnoli. S. , Turbine End wall Contouring Through Advanced Optimization Techniques Turbine end wall contouring, ASME Jnal of Turbomachinery, August 2023 vol 145

[55] Shabbir, A., Adamczyk, J.J., Strazisar, A.J., and Celestina, M.L., 2007, The Effect of >Hub Leakage on Two High Speed Axial Flow Compressor Rotors, ASME Paper No. 97-GT-346.

[56] Barrand, J.P., Caignaert, G., Graeser, J.E., and Rieutord, E., Synthesis of the results of tests in air and water aimed at detecting the critical recirculating flow rates on the inlet and outlet of a centrifugal pump impeller. La Houille Blanche, N°5 (August 1985), pp. 405-420.
https://doi.org/10.1051/lhb/1985028

[57] Stoffel, B., and Schuller, W., Investigations in Respect to the Determination of Cavitation Intensity by the Means of Acoustic Signals La Houille Blanche, N°7-8 (Décembre 1992), pp. 601-604. htpps://doi.org/10.105(/lhb/1992064

[58] Pelz, P.F., Keil, T., Ludwig, G. (2014). On the Kinematics of Sheet and Cloud Cavitation and Related Erosion. In: Kim, KH., Chahine, G., Franc, JP., Karimi, A. (eds) Advanced Experimental and Numerical Techniques for Cavitation Erosion Prediction. Fluid Mechanics and Its Applications, vol 106. Springer, Dordrecht. https://doi.org/10.1007/978-94-017-8539-6_9

[59] Hofmann, H., Stoffel, B., Coutier-Delgosha, O., Fortes-Patella, R., Reboud, J.L., Experimental and numerical studies on a centrifugal pump with 2D-curved blades in cavitating condition, Conference CAV2001:sessionB7.005, June 2001.

[60] Bakir, F., Kouidri, S., Noguera, S., Rey. R., Experimental Analysis of an Axial Inducer Influence of the Shape of the Blade Leading Edge on the Performances in Cavitating Regime, *J. Fluids Eng*. Mar 2003, 125(2): 293-301, https://doi.org/10.1115/1.1539872

[61] Dehnavi; E., Danlos. A., Solis, M., Kebdani, M., Bakir, F., Cavitation Performance of a Pump with Independently Rotating Inducer and Impeller in Co-Rotation and Counter-Rotation Modes by Varying Rotor Speeds, Journal of Fluids Engineering, June 2025, Vol. 147 / 061203 (15 pages)

[62] Zhang, L., Davila, G., and Zangeneh, M., Multi-Objective Optimization of a High Specific Speed Centrifugal Volute Pump Using Three-Dimensional Inverse Design Coupled with Computational Fluid Dynamics Simulations, *J. Fluids Eng*. Feb 2021, 143(2): 021202 (12 pages) Paper No: FE-20-1286 https://doi.org/10.1115/1.4048292

[63] Parikh, T., Mansour, M., and Thevenin, D., 2021, Maximizing the Performance of Pump Inducers using CFD-based Multi-Objective Optimization, Structural and Multidisciplinary Optimization, 65.9, htpps://doi.org/10.1007/s00158-21-03108-6.

[64] Gümmer, V., , Wenger, U., and Kau, H. P Using Sweep and Dihedral to Control Three-Dimensional Flows in Transonic Stators of Axial Compressors, ASME Journal of Turbomachinery, Jan 2001, Vol 123.

[65] Lopez, I. L., Shisu, T., Kipouros, T., Shapar, S., and Wilson, M., Extending Highly Loaded Axial Fan Operability Range Through Novel Blade Design, ASME, Journal of Turbomachinery, Dec. 2022, Vol. 144 / 121009-1

[66] Reutter, O., Hervé, S., Nicke, E., 2013, AUTOMATED OPTIMIZATION OF THE NON-AXISYMMETRIC HUB ENDWALL OF THE ROTOR OF AN AXIAL COMPRESSOR Proceedings of 10th European Conference on Turbomachinery Fluid dynamics & Thermodynamics ETC10, April 15-19, 2013, Lappeenranta,

[67] Leboeuf, F., Unsteady flow analysis in transonic turbine and compressor stages, VKI Lecture Series on Recent Developments in

Numerical Methods for Turbomachinery Flows, 2002-01)

[68] Moreau, S. and Roger, M., 2024, Turbomachinery Noise Review, International Journal of Turbomachinery and Power, Vol. 9,11, htpps://doi.org/10.3390/ijtpp9010011.
[69] He, L. 2021, Averaging for High Fidelity Modelling-Toward Large Eddy Simulations in Multi-Passage Multi-Row Configurations, Journal of Turbomachinery, Feb. 2021, Vol. 143

[70] Courtiade, N., Experimental analysis of the unsteady flow and instabilities in a high-speed multistage compressor, PhD Thesis, LMFA, Ecole Centrale de Lyon, Novembre 2012.

[71] Poensgen, C., Gallus, H.E., Three-Dimensional Wake Decay Inside of a Compressor Cascade and Its Influence on the Downstream Unsteady Flow Field: Part I—Wake Decay Characteristics in the Flow Passage, Journal of Turbomachinery. 113 (1991) 180.

[72] Mailach, R., Lehmann, I., Vogeler, K., Periodical Unsteady Flow Within a Rotor Blade Row of an Axial Compressor: Part II — Wake–Tip Clearance Vortex Interaction, in: Montreal, Canada, 2007: p. 1587-1597

[73] Ernst, M., Michel, A., Jeschke, P., Analysis of Rotor-Stator-Interaction and Blade-to-Blade Measurements in a Two Stage Axial Flow Compressor, J. Turbomach. 133 (2011) 011027.

[74] Courtiade, N., Ottavy, X., Modal decomposition for the analysis of the rotor-stator interactions in a high-speed multistage compressor, Journal of Thermal Science. 21 (2011)

[75] Roberts, Q. and Denton, J. D. Loss production in the wake of a simulated turbine blade. ASME paper 96-GT-421,1996.

[76] Cicatelli, G. and Sieverding, C. H. The effect of vortex shedding on the unsteady pressure distribution around the trailing edge of a turbine blade. ASME paper 96-GT-359, 1996.

[77] Tanga. E., Trebinjac. I., Ngo Boumb. G., Philita. M., 2017, Aerodynamic Interactions between a High-Pressure Turbine Stage and a Shroud Cavity. Paper ID: ETC2017-176. Proceedings of 12th European Conference on Turbomachinery Fluid dynamics & Thermodynamics ETC12, April 3-7, 2017; Stockholm, Sweden.

[78] Schulz, H. D., and Gallus, H. E., 1988. "Experimental Investigations of the Three-Dimensional Flow in an Annular Compressor Cascade". Journal Turbomachinery, 110(3), pp. 467-478.

[79] Schulz, H. D., Gallus, H. E., and Lakshminarayana, B., 1990. "Three-Dimensional Separated Flow Field in the Endwall Region of an Annular Compressor Cascade in the Presence of Rotor-Stator Interaction. Part I. Quasi-Steady Flow Field and Comparison with Steady-State Data". Journal of Turbomachinery, 112(4), pp. 669-678.

[80] Schulz, H. D., Gallus, H. E., and Lakshminarayana, B., 1990. "Three-Dimensional Separated Flow Field in the Endwall Region of an Annular Compressor Cascade in the Presence of Rotor-Stator Interaction. Part Ii. Unsteady Flow and Pressure Field". Journal of Turbomachinery, 112(4), pp. 679-690.

[81] Lei, V. M., Spakovsky. Z.S., and Greitzer. E. M. 2008. "A Criterion for Axial Compressor Hub Corner Stall." ASME Jnal of Turbomachinery, vol. 130, doi:10.1115/GT2006 91332.

[82] Xianjun Yu Baojie Liu, 2010 A Prediction Model for Corner Separation/Stall in Axial Compressors, Proceedings of ASME Turbo Expo 2010: Power for Land, Sea and Air GT2010, June 14-18, 2010, Glasgow, UK GT2010-22453.

[83] Jansen, W. (1964). Rotating stall in a radial vaneless diffuser. *Journal of Basic Engineering*, *86*, 750-758

[84] Fringe, P., & Van den Braembussche, R. (1985). A theoretical model for rotating stall in the vaneless diffuser of a centrifugal compressor. *Journal of Engineering for Gas Turbines and Power*, *107*(2), 507-513

[85] Moore, F. K. (1989). Weak rotating flow disturbances in a centrifugal compressor with a vaneless diffuser. *Journal of Turbomachinery*, *111*(4), 442-449

[86] Ferrara, G., Ferrari, L., & Baldassarre, L. (2004). Rotating stall in centrifugal compressor vaneless diffuser: experimental

analysis of geometrical parameters influence on phenomenon evolution. *International Jounal of Rotating Machinery*, *10*(6), 433-442. doi: 10.1080/10236210490503950.

[87] Ljevar, S., de Lange, H. C., & van Steenhoven, A. A. (2006). Two-dimensional rotating stall analysis in a wide vaneless diffuser. *International Journal of Rotating Machinery*, Article ID 56420, pp. 1-11. doi:10.115/IJRM/2006/56420

[88] Dazin, A., Coudert, S., Dupont, P., Caignaert, G., Bois, G. (2008). Rotating instability in the vaneless diffuser of a radial flow pump. *Journal of Thermal Science*, *17*(4), 368-374.

[89] Dazin, A., Cavazzini, G., Pavesi, G., Dupont, P., Coudert, S., Ardizzon, G., Caignaert, G., Bois, G. (2011). High-speed stereoscopic PIV study of rotating instabilities in a radial vaneless diffuser. *Experiments in Fluids*, *51*, 83-93.

[90] Day I.J. Stall, Surge, and 75 years of Research. Journal of Turbomachinery, January 2016, Vol 138.

[91] B. Hellmich, J.R. Seume, Causes of Acoustic Resonance in a High-Speed Axial Compressor, Journal of Turbomachinery. 130 (2008) 031003

[92] Francesco Romanò, Clemence Rannou, Alberto Baretter, Julien Marty, Pierric Joseph, Antoine Dazin, 2025, How inlet flow inhomogeneities can affect the transition to stall in an axial compressor?, Proceedings of the 16th European Turbomachinery Conference (ETC16) Turbomachinery, Fluid Dynamics and Thermodynamics, March 24-28, 2025, Hannover, Germany, Paper ID: ETC2025-304.

[93] Third International Symposium on Cavitation, April 1998, Grenoble, France TWO-PHASE FLOW STRUCTURE OF CAVITATION : EXPERIMENT AND MODELLING OF UNSTEADY EFFECTS Jean-Luc REBOUD, Benoit STUTZ*, Olivier COUTIER

[94] Callenaere, M., Franc, J.P., Michel, J.M., & Riondet, M. (2001). The cavitation instability induced by the development of a re-entrant jet. Journal of Fluid Mechanics, 444, 223–256.

[95] Leroux, J.B., Coutier-Delgosha, O., & Astolfi, J.A. (2005). A joint experimental and numerical study of mechanisms associated to instability of partial cavitation on two-dimensional hydrofoil. Physics of Fluids, 17, 052101.
[96] Schnerr, G.H., Sauer, J. (2001), Physical and numerical modelling of unsteady cavitation dynamics, Proc. of the Fourth International Conference on Multiphase Flow, New Orleans.

[97] R. Fortes-Patella, O. Coutier-Delgosha, J. Perrin and J.-L. Reboud, "A Numerical Model to Predict Unsteady Cavitating Flow Behavior in Inducer Blade Cascades," Journal of Fluid Engineering, Vol. 129, No. 2, 2007, pp. 128-135. doi:10.1115/1.2409320

[98] Vasquez, R., and Adami, P., 2019, Roles and Challenges on CFD in the Design of Gas Turbines, Private communication, Invited Lecture No.15 European Turbomachinery Conference, ETC 13.

[99] Hammond. J., Pepper. N., Montomoli. F., and Michelassi. V., Machine Learning Methods in CFD for Turbomachinery: A Review, International Journal of Turbomachinery Propulsion and Power · May 2022, DOI: 10.3390/ijtpp7020016

[100] Raina, G., Bousquet, Y., Luquet, Y., Lippinois, E., Binder, N., 2024, Loss Breakdown in Axial Turbines: A New Method for Vortex Loss and Wake Detection from 3D RANS Simulations, Paper GT2024-121181, Proceedings of ASME Turbo Expo 2024 Turbomachinery Technical Conference and Exposition GT2024 June 24-28, London, United Kingdom

[101] Raina, G., Bousquet, Y., Luquet, Y., Lippinois, E., Binder, N., 2025, Analysis of Interactions among loss-generating mechanisms in axial flow turbines. . Proceedings of the 16th European Turbomachinery Conference (ETC 16) Fluid Dynamics and Thermodynamics, March 24-28, 2025, Hannover , Germany, Paper OD: ETC2025-150
.
[102] Bergmann, M., Morsbach, C., Möller, F. M., Goinis, G., Klose, B., 2025, A comparative study of varying incidence angle effects on a low-Reynolds number compressor cascade based on experiments, low fidelity and high-fidelity numerical simulations. Proceedings of the 16th European Turbomachinery Conference (ETC 16) Fluid Dynamics and

Thermodynamics, March 24-28, 2025, Hannover, Germany, Paper ID: ETC2025-231

[103] Yamade, Y., Kato, C., Nagahara, T., and Matsui, J., 2019, Origin and formation process of a submerged vortex in a pump sump, IOP Conf. Ser.: Earth Environ. Sci. 240 032028

[104] Bicchi. M., Biliotti. D. , Marconcini. M., Toni. L. , Cangioli. F., and Arnone. A., 2022, An AI-Based Fast Design Method for New Centrifugal Compressor Families. MDPI *Machines* 2022, *10*, 458. https://doi.org/10.3390/machines10060458

[105] Ottavy, X., Brandstetter, C., Workshop-Open test case fan ECL5/CATANA, Introduction and Presentation of the tet case, . Proceedings of the 16th European Turbomachinery Conference (ETC 16) Fluid Dynamics and Thermodynamics, March 24-28, 2025, Hannover , Germany,